Simulation model of vertical water wheel performance flow

by Eko Yohanes Setyawan Eko Yohanes Setyawan

Simulation model of vertical water wheel performance flow

View the article online for updates and enhancements.

Simulation model of vertical water wheel performance flow

Eko Yohanes Setyawan[1*], Soeparno Djiwo[1], Djoko Hari Praswanto[1], Purbo Suwandono[2], Parulian Siagian[3], Waldemar Naibaho[3]

[1]Mechanical Engineering, Faculty of Engineering, National Institute of Technology Malang, jl. Bendungan Sigura-gura No.2 Malang 65152, Indonesia
[2]Mechanical Engineering Department, Universitas Widyagama Malang, Jl. Taman Borobudur, Malang, Indonesia
[3]Mechanical Engineering Department, Universitas HKBP Nommensen, Medan

[*]yohanes@lecturer.itn.ac.id

Abstract. Potential renewable energy source that can be greatly utilized is the water flow, namely hydropower. The hydropower is one of the efforts to answer the energy crisis occurred in several regions due to lack of awareness to develop alternative energy. This effort will be done by designing a hydroelectric power plant. Before being made, it was studied using CFD simulation using *Ansys Student Version 19.1* that was calculated based on flow assumptions. One model of the approach used is *k-epsilon*, with the initial flow rate of 5 m/s. By using water-liquid fluid material, the speed of water produced around the wheel is 2.19 m/s while the speed of water after passing the water wheel is 5.07m/s.

1. Introduction

There are number of regions in Indonesia which have the potential of renewable energy sources, one of which is hydropower. The energy crisis that occurred in several regions was due to the lack of awareness to develop additional plants fueled by renewable energy. Based on the issue of the crisis of electricity and the increasing energy needs, the energy potential of Indonesian rivers must be utilized to the greatest extent possible. One of them is by designing a hydroelectric power plant [1,2] to meet electricity needs by utilizing small water flow sources [3,4].

The process of making waterwheels need to spend a lot of money in order to produce maximum performance. If it is using the material around, the efficiency result will be quite low, reaching 24% [5] because it is done traditionally. In order to increase the efficiency up to 60%, a classic vertical type of water wheel can be used [6,7] by working on the change in potential energy in the flow of water into mechanical energy by the water wheel. The second change is the mechanical energy produced. This will be used to rotate a generator that will produce the electrical energy.

Prasad et al. [8] compared the axial efficiency of flow turbines comes through experimental and CFD analysis with three different propeller guide angles. Jain and Saini [9] have estimated the performance and efficiency of Francis runners in 4 different vane guide point operations using CFDs and to validate the same as model testing. Kim et al. [10] predicted the performance of tubular hydro-type turbines for openings guide variable openings and the effects of pressure, tangential and axial studied velocity distribution on turbine performance using CFD [11].

The design of the water wheel is expected to be able to fulfill small-scale power consumption especially for household consumers in areas far from the city. For this reason, a

power plant that is designed utilizes river water flow energy that has medium flow capacity which is often found in regions. Therefore, the purpose of this paper is to study the flow characteristics of vertical waterwheels designed and simulated using CFDs.

2. Method

This research method uses analysis by collecting various references relating to waterwheels. Then make a numerical simulation design of waterwheel using Ansys Student Version 19.1. Input of water turbine with dimensions of blade length 600 mm and width of blade 100 mm number of blades 12 pieces, obtained from the calculation below:

$$S = L \times (D/2) \times (1 - \cos\theta / \sin \alpha) \tag{1}$$

S is a submerged blade surface where L is the width of the water wheel (m), D is the diameter of the water wheel (m) and α is the slope angle.

$$P = Cp \times (\rho/2) \times vr^2 \times S \tag{2}$$

P is the lift force; where Cp lift coefficient; ρ is the density (kg/m3); Vr is the radial velocity (m / s); S is a submerged blade surface (m)

$$R = Cp \times (\rho/2) \times vt^2 \times S \tag{3}$$

The tensile force where Cp is the lift coefficient ρ is the density (kg / m3); Vt is tangential speed (m/s); S is a submerged blade surface (m)

$$F = \sqrt{P^2 + R^2} \tag{4}$$

F is the resultant force.

$$Fu = F \times \cos \varepsilon \tag{5}$$
$$\varepsilon = \arctan (CR/CP) - (a - \gamma) \tag{6}$$

FU is a useful force and its angle, where γ is the relative velocity angle.

$$M = Fu \times (D/4) \times (1 + \cos\theta / \sin \alpha) \tag{7}$$

M is the useful moment, where θ is the angle between the center of the windmill and the position of the maximum load blade.

$$Nu = M \times \omega = Fu \times vt \tag{8}$$
$$Nmed = Ki \, S(Nu/n) \tag{9}$$

Nu is momentary strength and average strength where n is the number of count points between zero position of action, the value is usually taken between $1 < n < 6$ and the position of maximum blade load, while Ki is the active blade coefficient, for Ki turbine undershot type = 1, 6.

$$Nmax = (\rho/2) \times S \, max \times vam^3 \tag{10}$$
$$Smax = L \tag{11}$$

$Nmax$ is the maximum strength of the flow, $Smax$ is the maximum surface of the submerged blade and h is the submerged blade ratio.

$$\eta = Nmed / Nmax \tag{12}$$

η is the efficiency of the waterwheel. Whereas ω isangular speed:

$$\omega = 4vt / [D(1 + \cos\theta / \sin \alpha)] \tag{13}$$

$n \, med$ rotation speed:

$$n \, med = (30/\pi n) \times S \, \omega j \tag{14}$$

$$\theta = cos\text{-}1[\{(0.5 \; x \; D) - (B \; x \; h)\}/ \; (0.5 \; x \; D)] \tag{15}$$

$$\alpha = (90 - \theta \,) + \theta \; x \; (m \,/ \, m{+}1) \tag{16}$$

m is a constant of $1 \le m \le 6$

To find out the Radial speed, the equation below is used:

$$Vr = \surd(\, Vam^2 - 2 \; x \; Vam \; x \; Vt \; x \; sin \; a + Vt^2) \tag{17}$$

Tangential velocity used

$$Vt \; = (Vam \; x \; sin \; \alpha) \; x \; 0.5 \; x \; [1 + (\, cos\theta / sin \; \alpha)] \tag{18}$$

$$tan \; \gamma = (Vt \; x \; cosa) \, / \, (Vam - Vt \; x \; sin \; a) \tag{19}$$
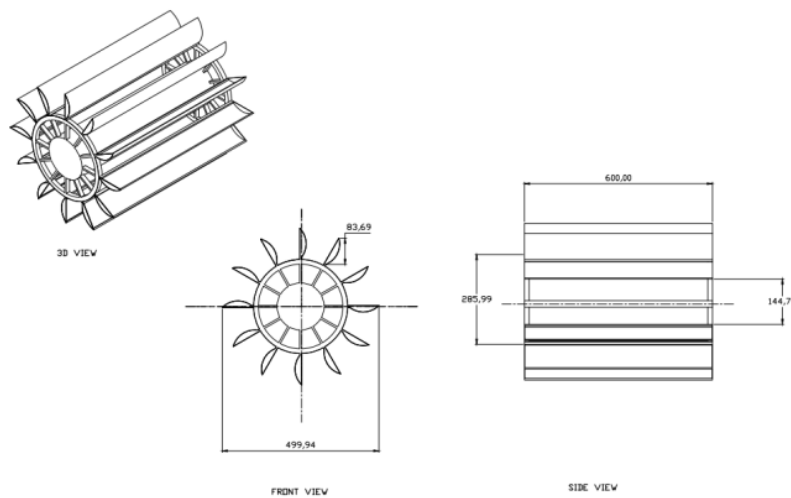

Figure 1. Design and dimensions of a waterwheel simulation area

The design and dimensions in Figure 1. above are obtained from the calculation of 1 to 15. In accordance with the purpose of this study, a waterwheel planning was made which was later simulated using CFD. CFD is a method of numerical fluid flow approach with the help of a computer, which can work in low water flows. This condition is suitable with the character of the river flow in the regions therefore it can be applied in Indonesia with many small islands. The vertical waterwheel design and construction obtained as shown in Figure 1, dimensions of blade length 600 mm and width of blade 100 mm with number of blades 12 pieces, to determine the speed of water flow before and after passing through the waterwheel.

3. Results and Discussions
3.1. Initial and boundary conditions
Waterwheel modelling using Ansys Student Version 19.1 is calculated based on flow assumptions, with an initial flow rate of 4.85 m/s by entering the gravity acceleration value 9.81 m/s2. One of the models used is *k-epsilon*. Modelling by using this system provides advantages in the form of computational power efficiency, stability of numerical calculations and accuracy of the resulting solution. By using water-liquid fluid material, stating that the waterwheel moves based on water flow by selecting "Relative to cell zone: fluid-solid" means that the speed of this wheel will be adjusted to the conditions in the water flow.

3.2. Simulation results

Numerical analysis of the water wheel assumed that the waterwheel was half floating on the surface of the water, in accordance with the limitation of the problem that the incoming water flowing at a speed of 5 m/s then from the flow moves the windmill. The flow rate of the water that hit the blade on the water wheel causes the water wheel to rotate as shown in Figure 2 and Figure 3. In Figure 2 the red color is a waterwheel that gets pressure from the flow of water with a number of 12 pieces of blade. Where from the simulation results the pressure distribution experienced by each water wheel blade is evenly distributed, shown from the yellow color of each water wheel blade with an even pressure value of 43738.4 Pa.
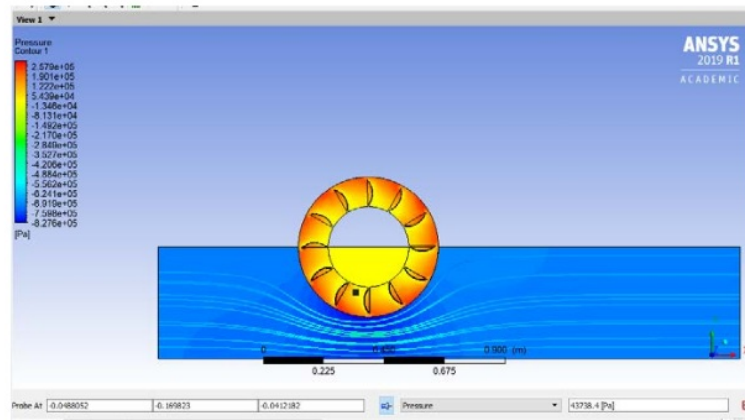

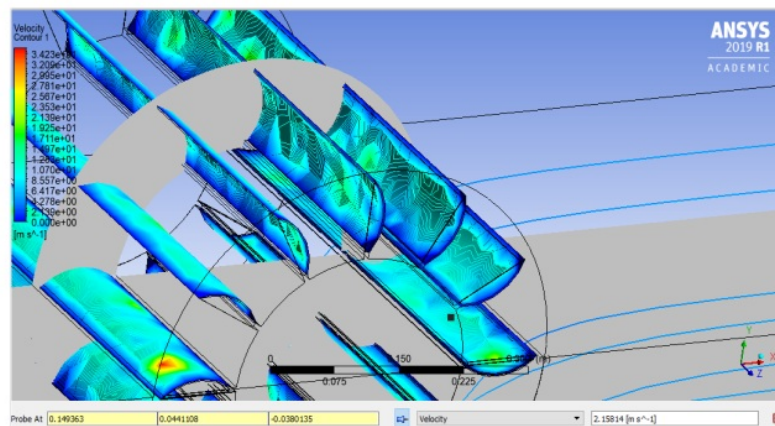**Figure 2.** Pressure experienced by a mill on a water wheel

Figure 3. Speed experienced by a water wheel when spinning

Figure 3 shows the results of the velocity distribution of water flowing on each blade of the waterwheel. The red color shows the highest flow velocity area on the waterwheel blade

with a maximum value of 3.42 m/s, while the lowest water flow velocity that affects the waterwheel blade is light blue with a value of 2.158 m/s.
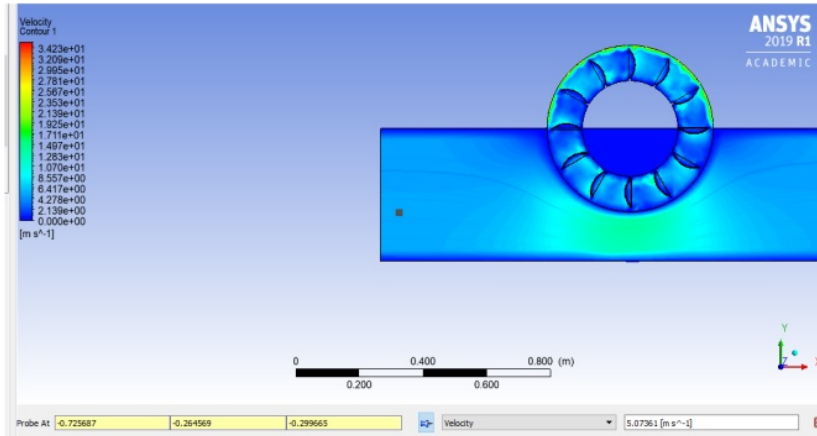

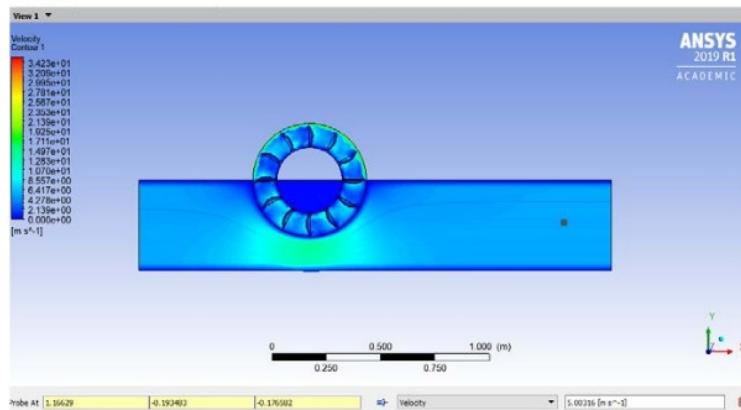**Figure 4.** Initial water flow in the waterwheel

Figure 5. Flow velocity after passing the water wheel

The results of numerical analysis show that from the simulation results of the waterwheel obtained at 2.19 m/s, the speed of water flow after passing the water wheel shown in Figure 5 is 5.076m/s, and the maximum pressure experienced by each blade of water is 42326 Pa.

4. Conclusion

The phenomenon of water flow in the waterwheel has been studied using CFD simulation using *Ansys Student Version 19.1* calculated based on flow assumptions, with an initial flow velocity of 5.00 m/s by entering the gravity acceleration value 9.81 m/s2. The position of the water wheel is assumed to be half float. The model of approach used is *k-epsilon*. By using water-liquid fluid material, stating that the waterwheel moves based on the flow of water by

selecting "Relative to cell zone: fluid-solid". The speed of water produced around the mill is 2.19 m/s while the speed of water after passing the water mill is 5.07m/s with the pressure experienced by the waterwheel is 42326 Pa.

References

[1].   Panwar NL, Kaushik SC, Surendra K. Role of renewable energy sources in environmental protection: a review. Renewable and Sustainable Energy Reviews 2011;15:1513–24.

[2].   Yuksek O, Komurcu MI, Yuksel I, Kaygusuz K. The role of hydropower in meeting Turkey's electric energy demand. Energy Policy 2006;34:3093–103.

[3].   Yuksel I. Hydropower in Turkey for a clean and sustainable energy future. Renewable and Sustainable Energy Reviews 2008;12:1622–40.

[4].   Yukse, I "As a renewable energy hydropower for sustainable development in Turkey" Renewableand Sustainable Energy Reviews 14 (2010) 3213–3219

[5].   Vashisht AK. Current status of the traditional watermills of the Himalayanregion and the need of technical improvements for increasing their energyeffiency. Appl Energy 2012;98:307-15.

[6].   Miller G, Kauppert K. Performance characteristics of water wheels. J HydraulRes 2004;42(5):45-60.

[7].   Denny M. The effiency of overshot and undershot waterwheels. Eur J Phys2004;25:193-202.

[8].   Prasad, V, Gahlot, V K & Krinnamachar, P "CFD approach for design optimization and validation for axial flow hydraulic turbine" Indian Journal of Engineering & Material secince Vol. 16, August 2009,pp 229-236

[9].   Jain, S, Saini,R P & Kumar, A "CFD approach for prediction of efficiency of Francis turbine" IGHEM-2010, oct. 21-23, AHEC, IIT Roorkee, India

[10].   Kim, Y T, Nam, S H, Cho, Y J, Hwang, Y, C, Choi,Y D, Nam, C D & Lee, Y H "Tubular-Type hydro turbine performance for variable guide vane opening by CFD" The fifth international conference on fluid mechanics, Aug 15-19,2007, Shanghai, Chaina.

[11].   E.Y Setyawan, S Djiwo and T Sugiarto, Simulation Model of Fluid Flow and Temperature Distribution in Porous Media Using Cylinder Convergent and Divergent Nozzle, Internasional Journal of Technology and Sciences, 1 (2017) 1-10.

[12].   Setyawan, E. and Ambarita, H. (2018) 'A Preliminary Field Test of a Natural Vacuum Solar Desalination Unit Using Hybrid Solar Collector', in AIP Conference Proceedings. bali: American Institute of Physics. doi: 10.1063/1.5046598.

[13].   Setyawan, E. and Napitupulu, F. H. (2018) 'Study on the Characteristics of a Natural Vacuum Desalination System Using Solar Energy as Energy Sources', International Journal of Mechanical and Mechatronics Engineering, 18(05).

[14].   E.Y. Setyawan, R.A.M. Napitupulu, P. Siagian, H Ambarita, Field tests of a natural vacuum solar desalination system using hybrid solar collector, IOP Conf. Series: Materials Science and Engineering 237, (2017) 012012.

Simulation model of vertical water wheel performance flow